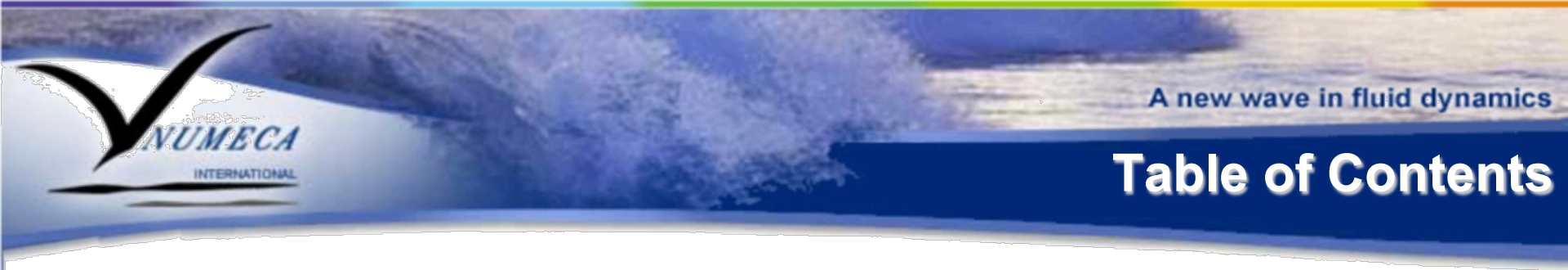




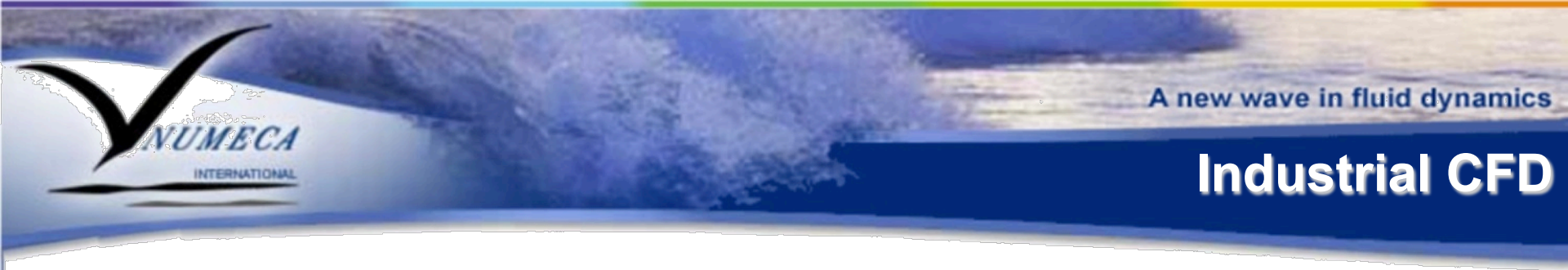
# THE CHALLENGES OF PRESENT AND FUTURE INDUSTRIAL CFD

Charles Hirsch  
Prof. Em. Vrije Universiteit Brussel  
President, NUMECA int.



# Table of Contents

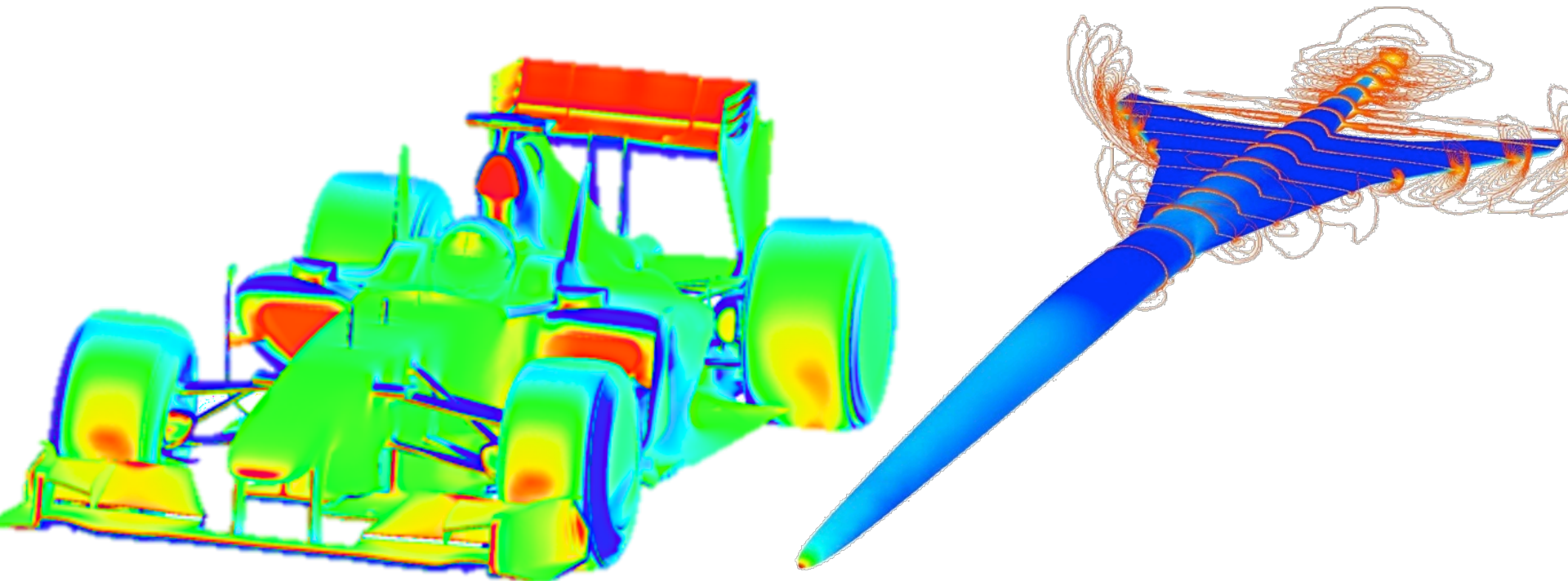
- CFD methods in industry
- Challenges in Grid Generation
- Challenges in CFD methods
- Other methods:
  - Non-Linear Harmonics (NLH)
  - High-Order methods (HOM)
  - Uncertainties Quantification (UQ)



## **Industrial CFD can be characterized by the following requirements and challenges**

- Diversity of flow configurations and applications
  - Aerospace; marine; engines; power generation; chemical process industry; combustion; .....
- Complexity of geometries and physics
- CFD Code requirements
  - From incompressible to hypersonics
  - From perfect or real gases to liquids and condensable fluids defined by thermodynamic tables
  - Reliability and robustness for all flow conditions
- **A few examples**

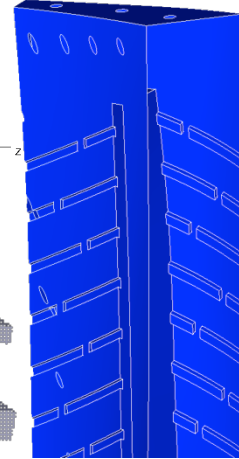
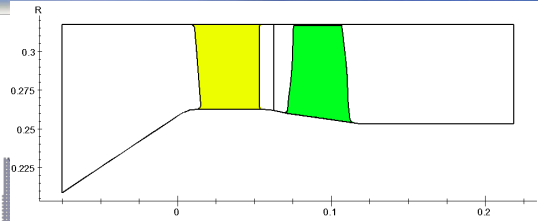
- Low speed flow around F1 car
- Supersonic flow around advanced aircraft configuration



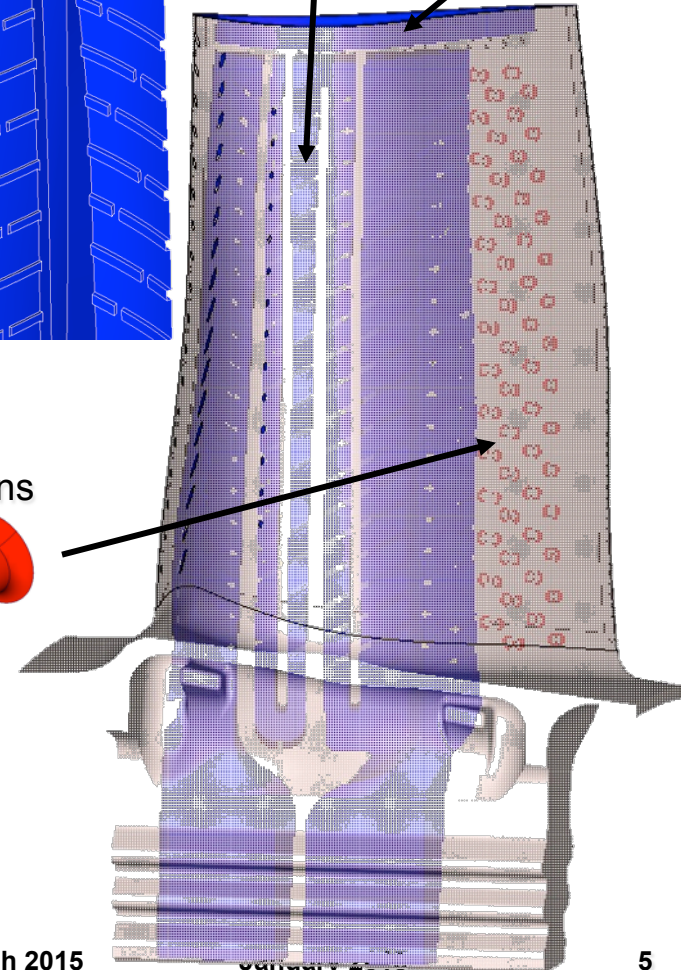


# High pressure cooled gas turbine stage

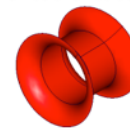
External Cooling holes  
Turbulence promoters



Ribbed channels  
Bassin



Pin Fins



Trailing edge bleed slots

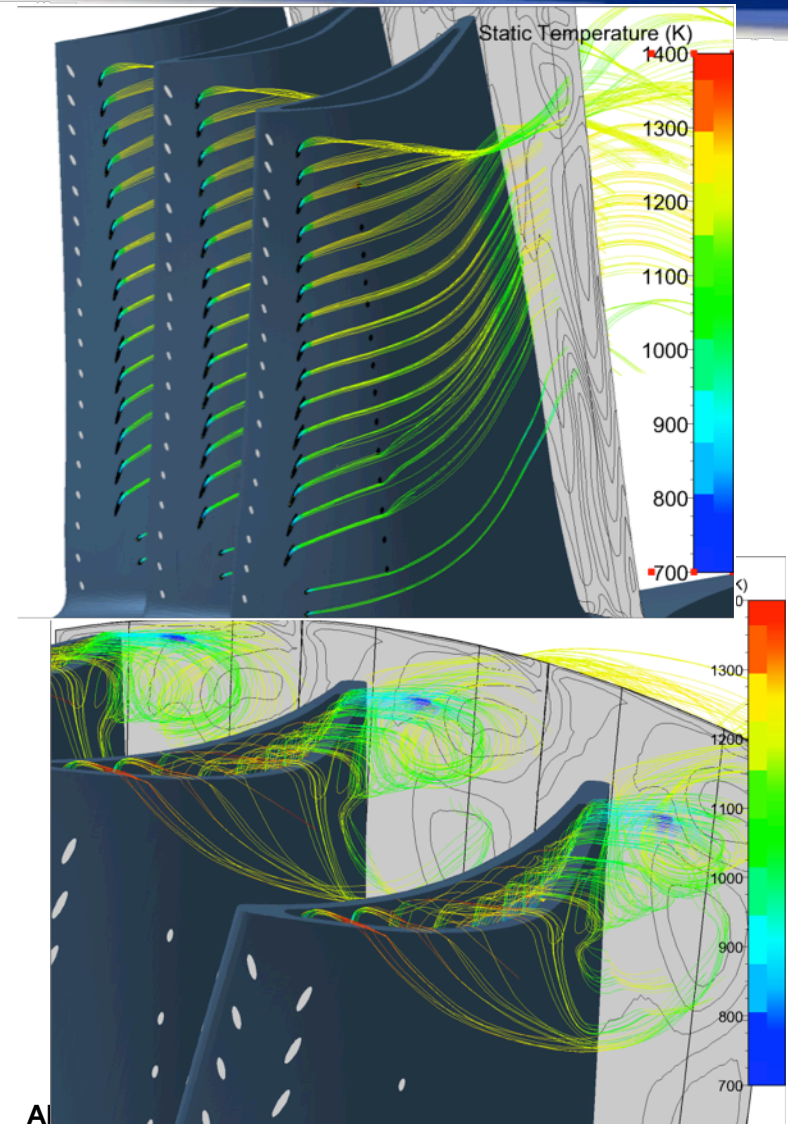
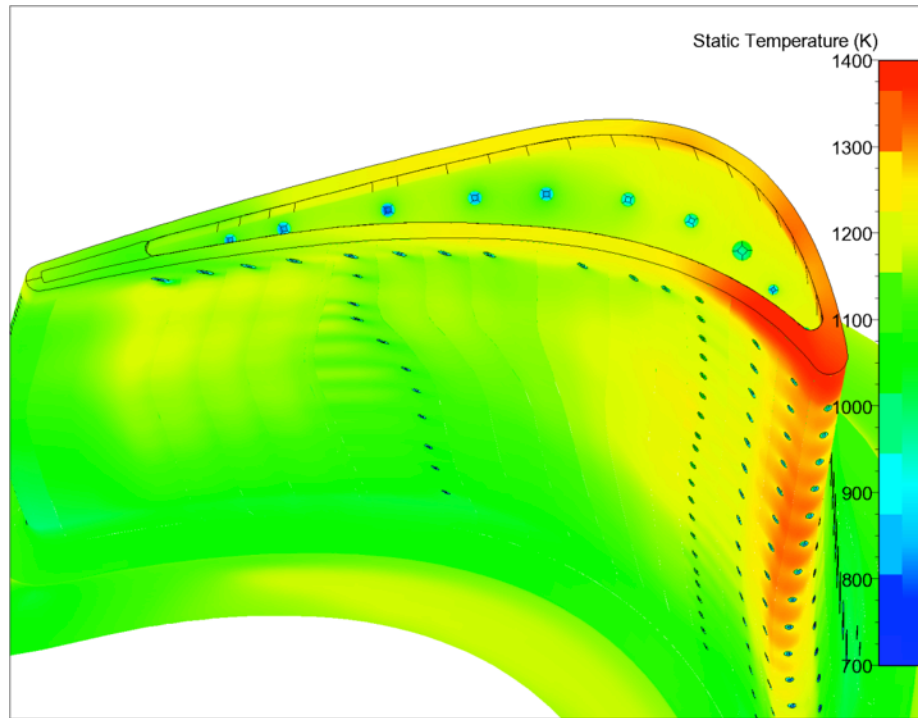
Insert 1

Insert 2

Impinging jet

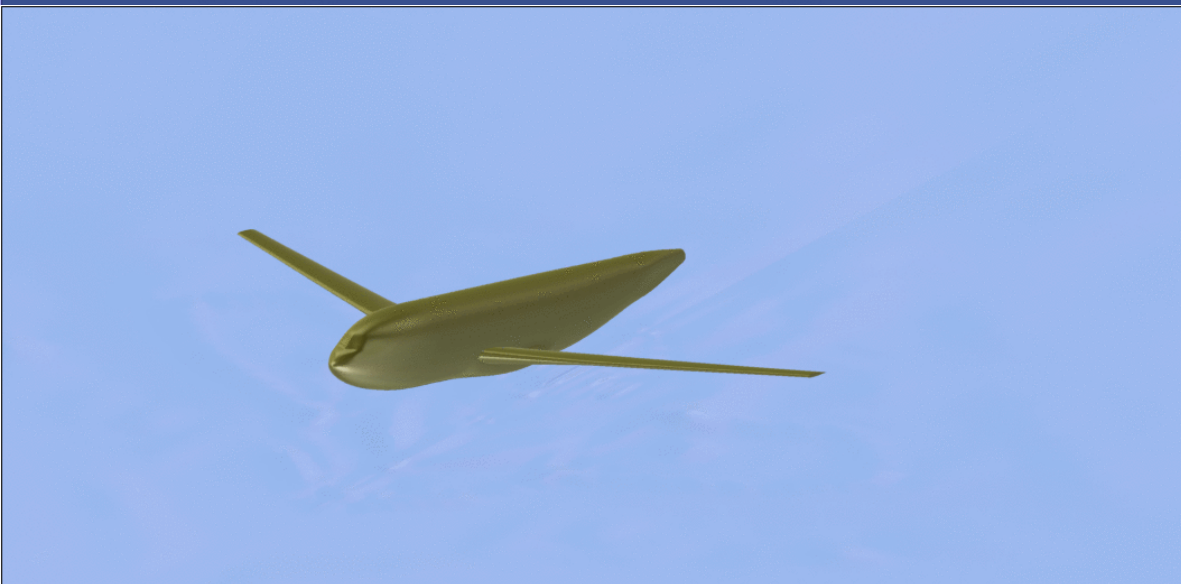


# Conjugate heat transfer (CHT) analysis of the HPT Stage





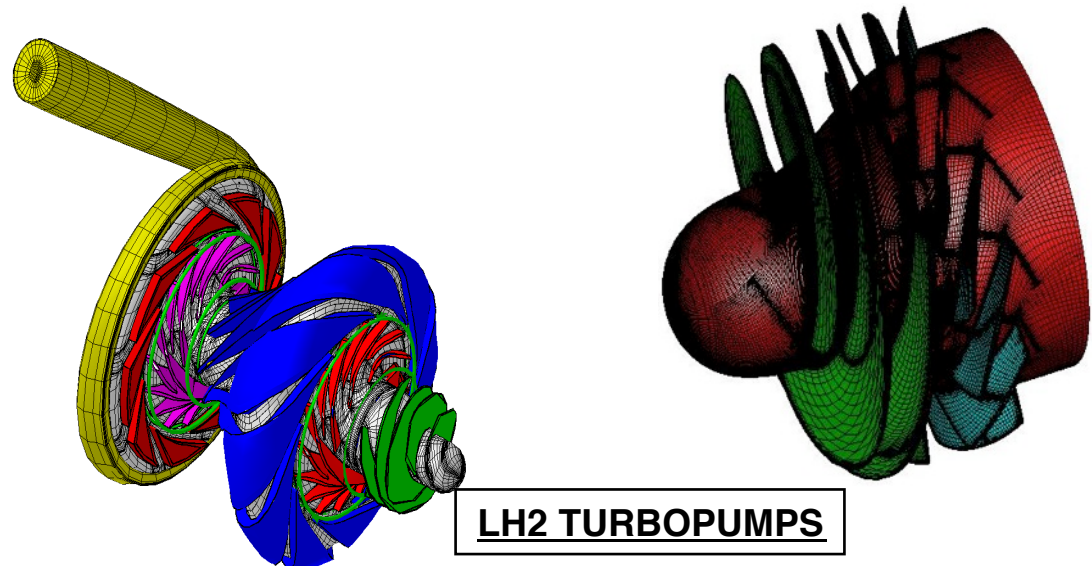
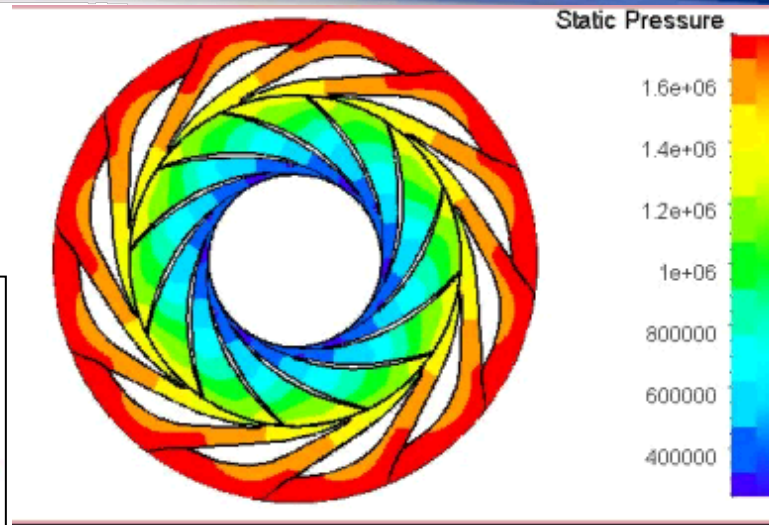
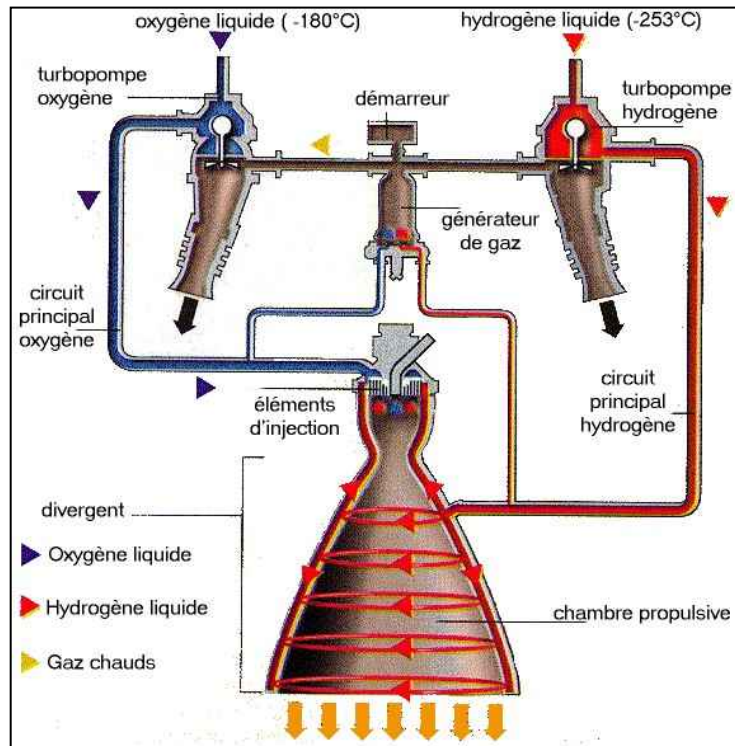
- Free surface and 6 DOF simulation for the prediction of hydrodynamic forces during the plane ditching





# Liquid Hydrogen Space pumps

## Cryogenic engines of European ARIANE 5 Space launcher





## Key issues

### **Four major challenges face the future of industrial CFD based analysis and design software systems:**

#### **(i) The requirements for High fidelity CFD, implying**

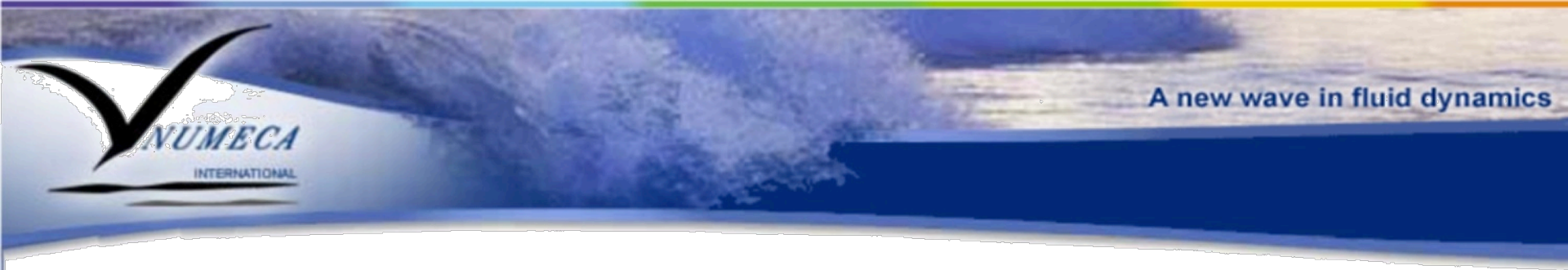
- More efficient mesh generation systems to accompany the continuous growth of the geometrical complexity
- Faster, more robust and accurate CFD algorithms, taking advantage of computer massively parallel hardware performance
- Fast methods for unsteady 3D flows, such as multistage rotating machines, based on Reduced Order Modeling methodologies
- Efficient methods for Multi-physics, such as fluid-thermal (CHT); fluid-structure (FSI); aero-acoustic couplings (CAA)

#### **(ii) the necessity for improvements in physical modeling, in particular turbulence and combustion models**

#### **(iii) the necessity to incorporate the uncertainties associated to numerical simulations, such as operational and geometrical uncertainties. This has a major impact on the design process in order to reduce the risks associated with the simulation based decision process**

#### **(iv) the development of robust design methodologies taking into account the presence of uncertainties**

**The key objective is the incorporation of powerful CFD solutions into MDO systems**



# ***High-Fidelity Grid Generation***

## Grid generation requirements

- **Grid generation is a major problem in the CFD process and is the bottleneck with complex geometries**
  - Can be very costly in engineer-time, unless it is made automatic
- **The quality of the simulation depends strongly on the grid quality, in terms of**
  - Smoothness
  - Regularity
  - Distortion of the cells
- **To start the grid generation process, an adequate definition of the geometry should be available**
  - This is provided by the CAD system, but
  - It is often very badly defined and requires many lengthy (and costly) manipulations

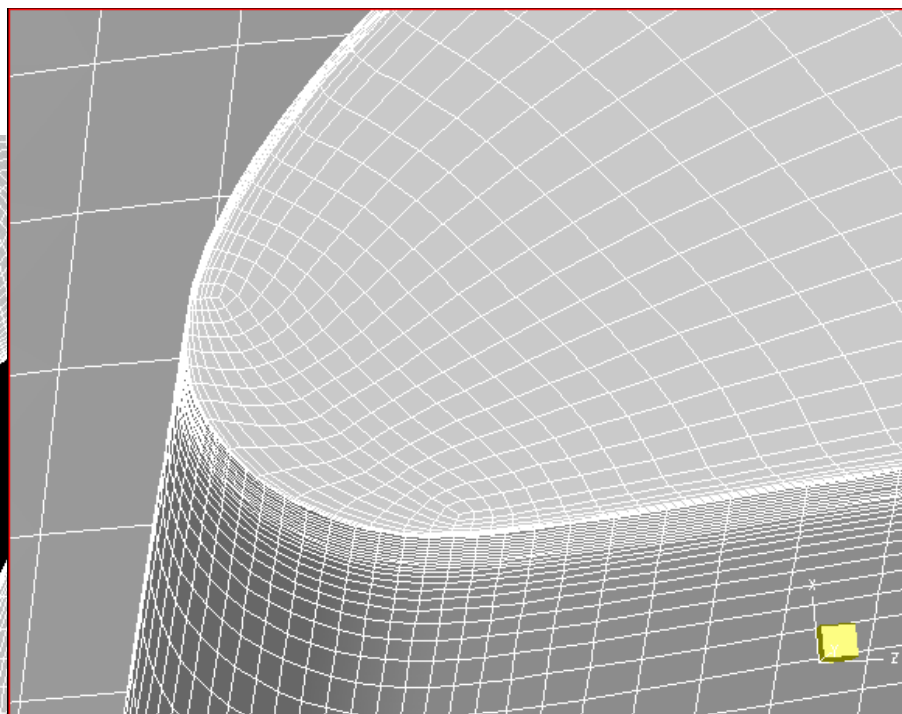
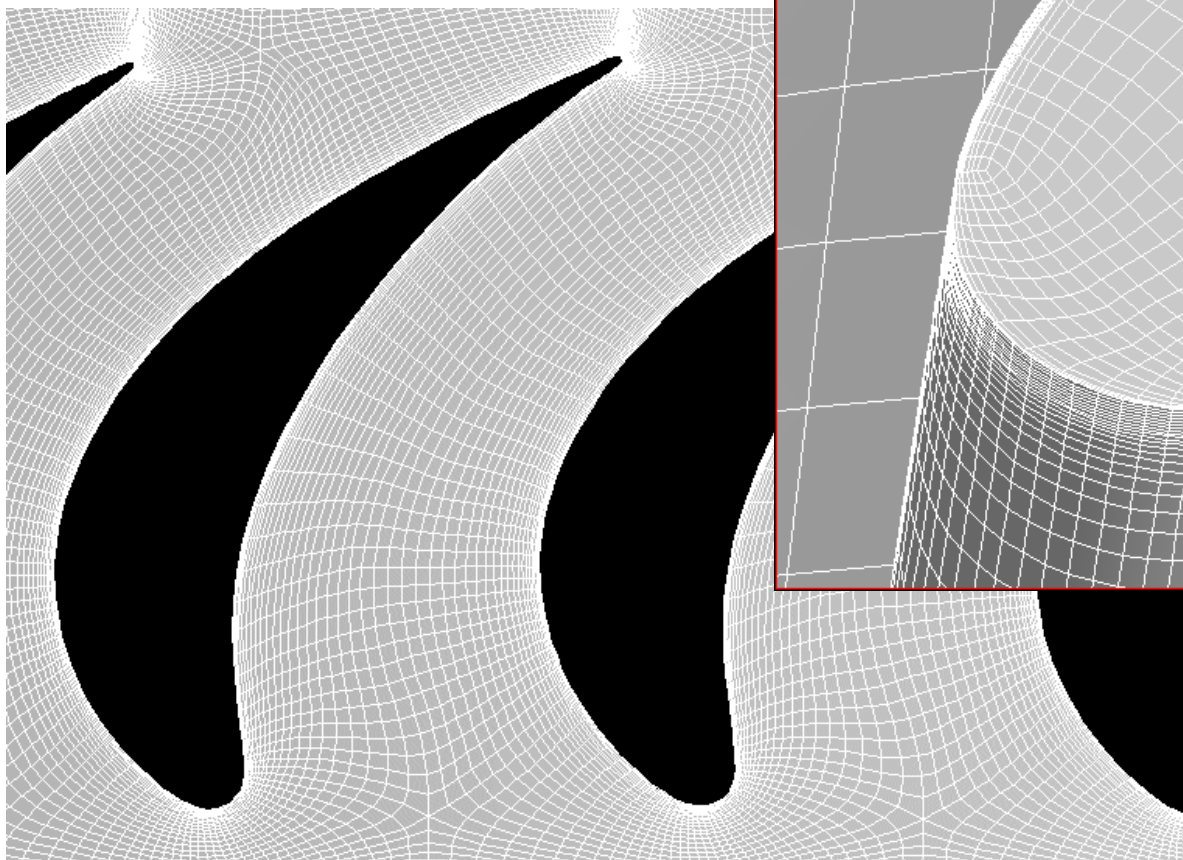
# Unstructured vs. structured techniques

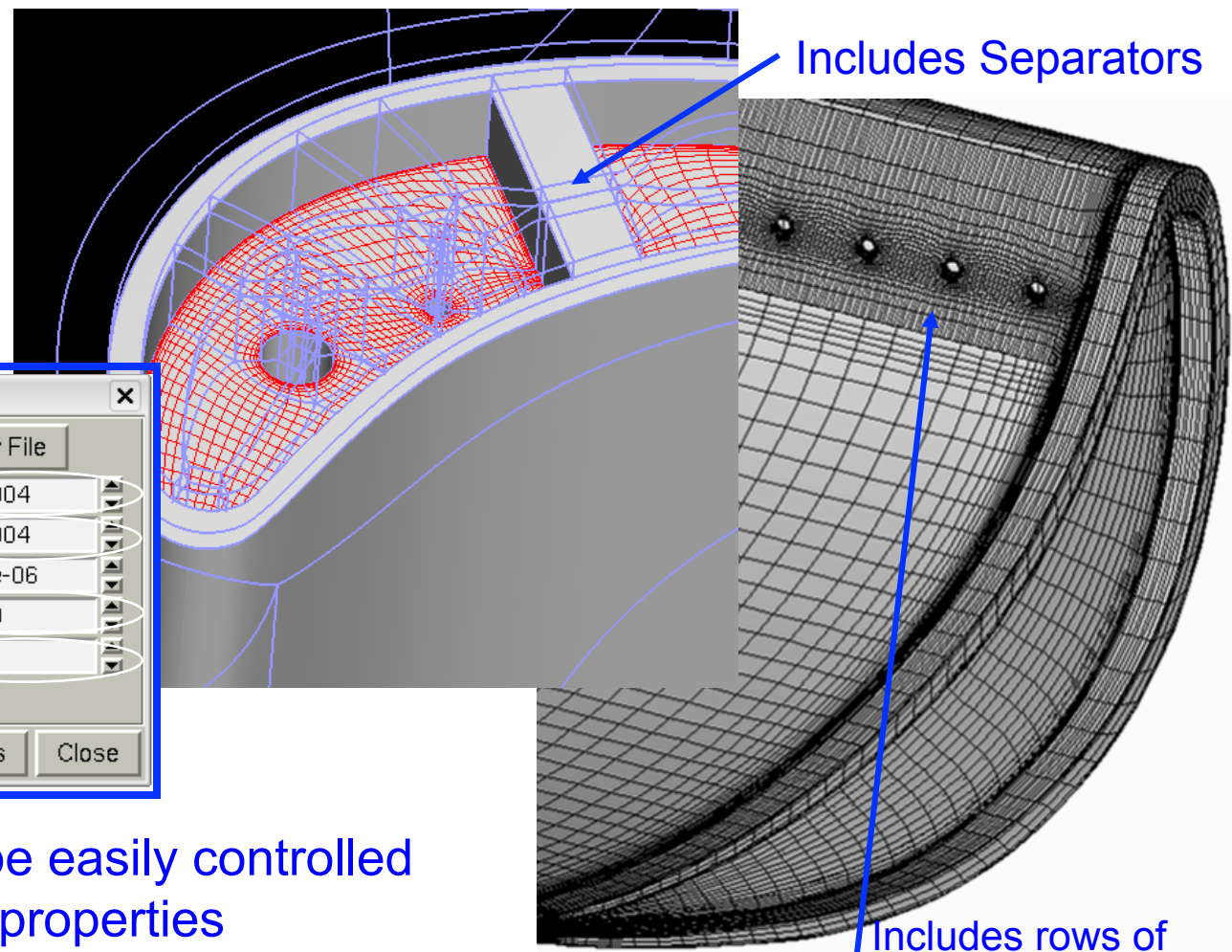
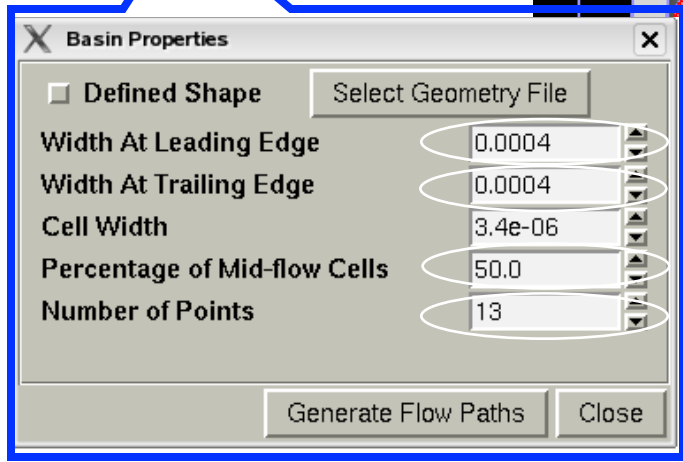
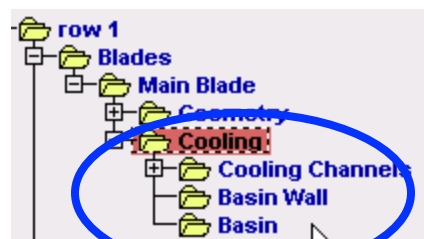
- **Structured, body aligned grids are the optimum choice for CFD**
  - Higher accuracy
  - Faster solvers
  - Less memory
- **The 'stiffness' of structured grids leads to severe constraints on the grid generation**
  - time consuming and hence expensive task
  - can be partly reduced via structured, multiblock grids with non-matching boundaries
- **However, we believe that structured grids should be applied whenever automatic grid generation is possible**
- **KEY ISSUE: CREATE AUTOMATIC STRUCTURED GRID GENERATION TOOLS FOR FAMILIES OF TOPOLOGIES,**
- **FOR INSTANCE FOR TURBOMACHINERY PASSAGES: Autogrid5 which has become the industry standard**





## Turbine Grid



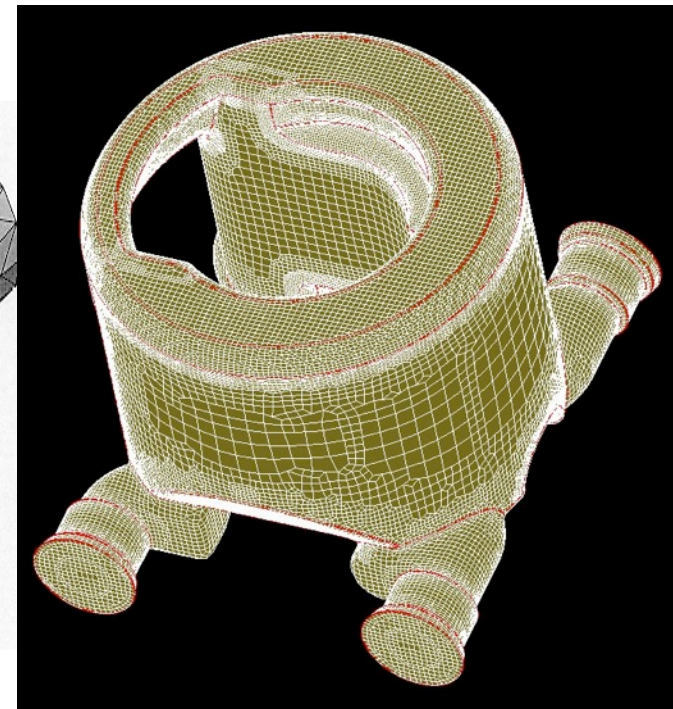
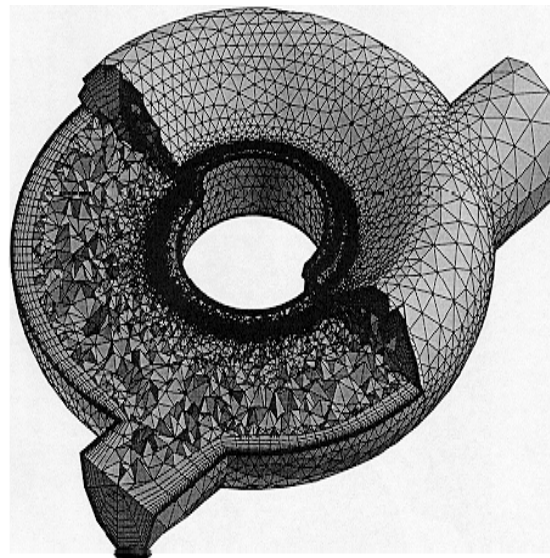
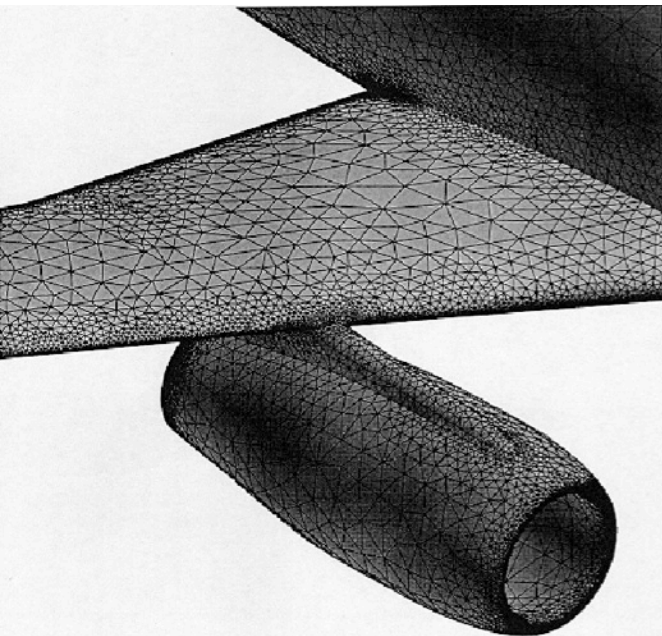


Basin Depth can be easily controlled as mesh properties



# Unstructured grids

- **Unavoidable for general complex geometries**
- **Various options are currently available**
  - Tetrahedral grids
  - Hexahedral grids
  - Hybrid grids





# Unstructured Hexahedral mesh generation

## Two options:

### • **HEXPRESS**

- Full unstructured meshes with hanging nodes (non-conformal)
- From volume to surface process, starting from an octree Cartesian mesh process

### • **HEXPRESS/Hybrid**

- Hex-dominant, from volume to surface, without hanging nodes
- Combination of tets, prisms and pyramids
- CAD cleaning during mesh generation process, with “hole searcher” methodology

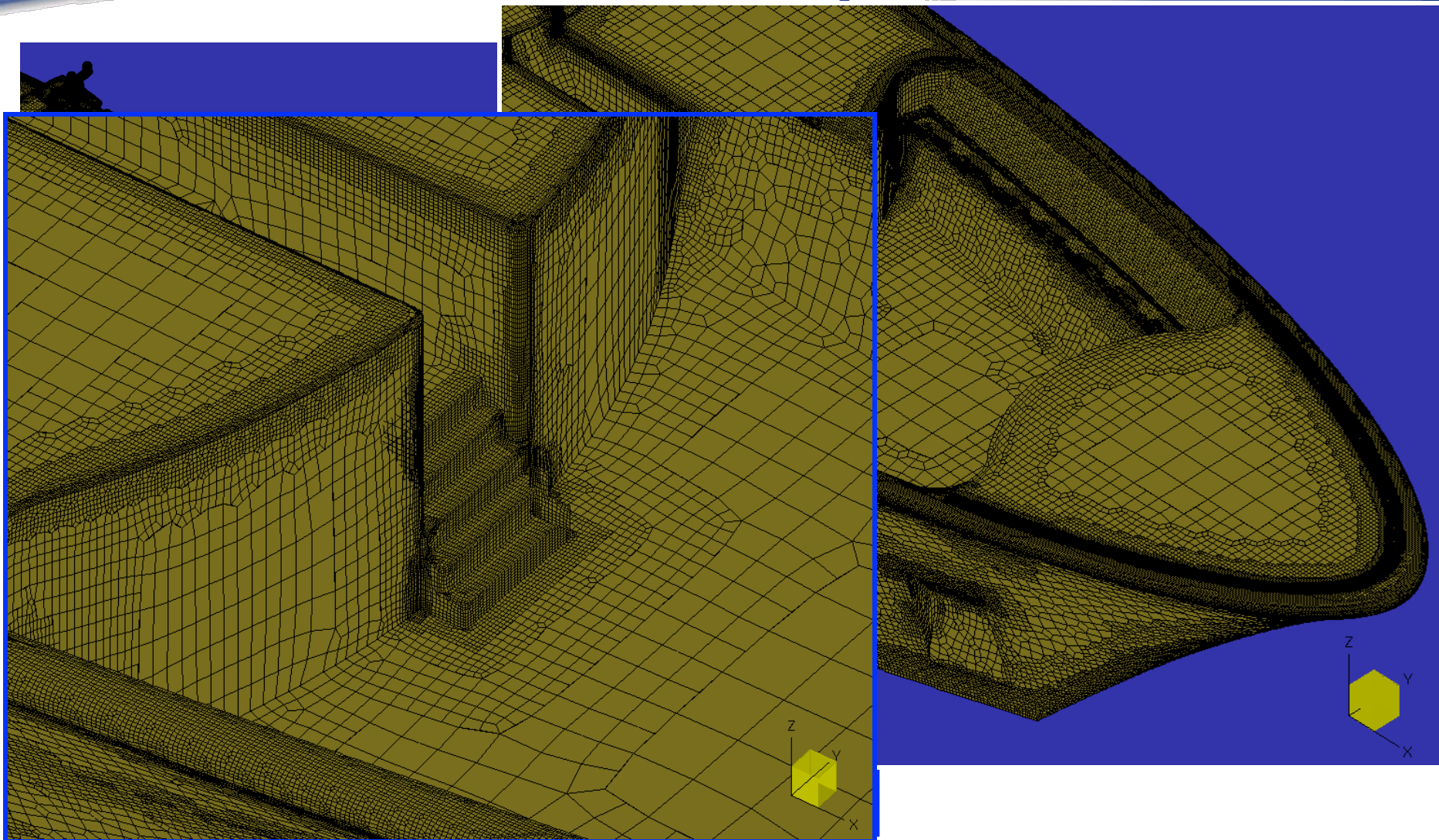
### • **A few examples**





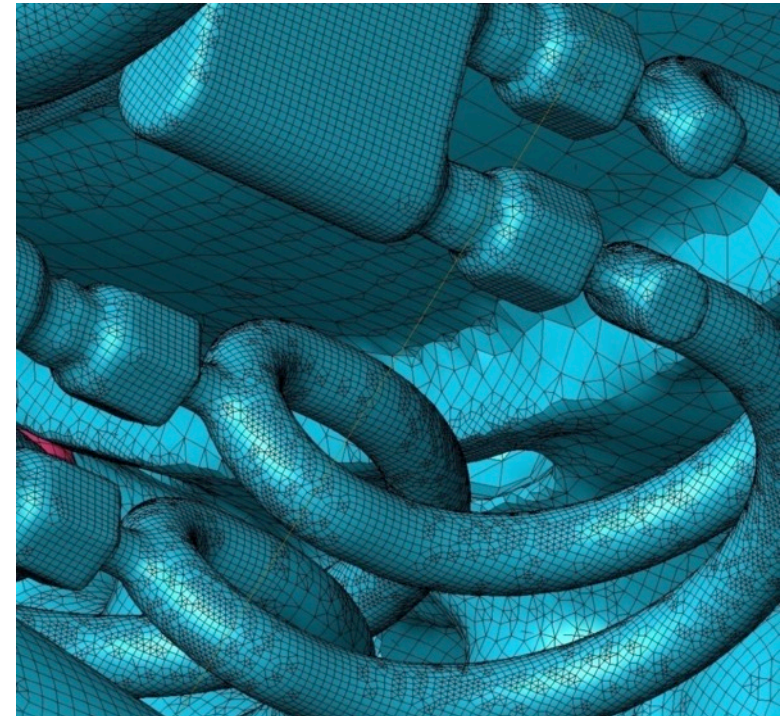
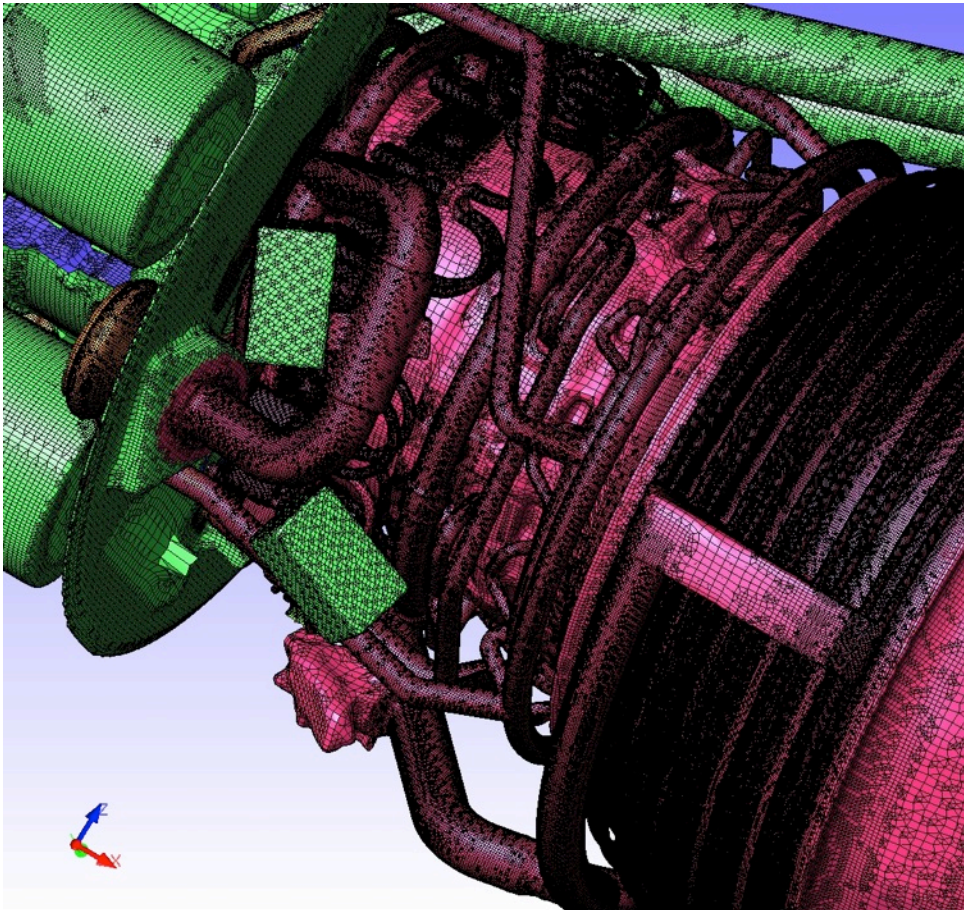
A new wave in fluid dynamics

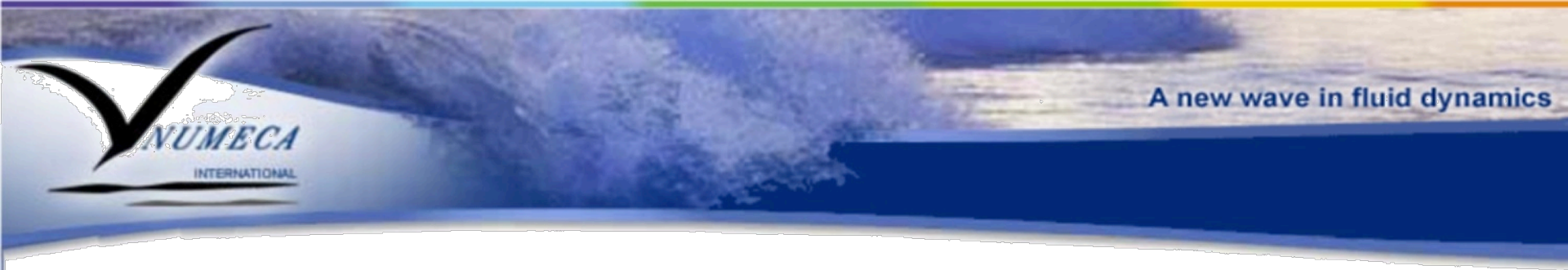
# Hexpress mesh on a Yacht





# Hexpress/hybrid mesh of an engine ventilation system





# ***High-Fidelity CFD methods***





## Main requirements:

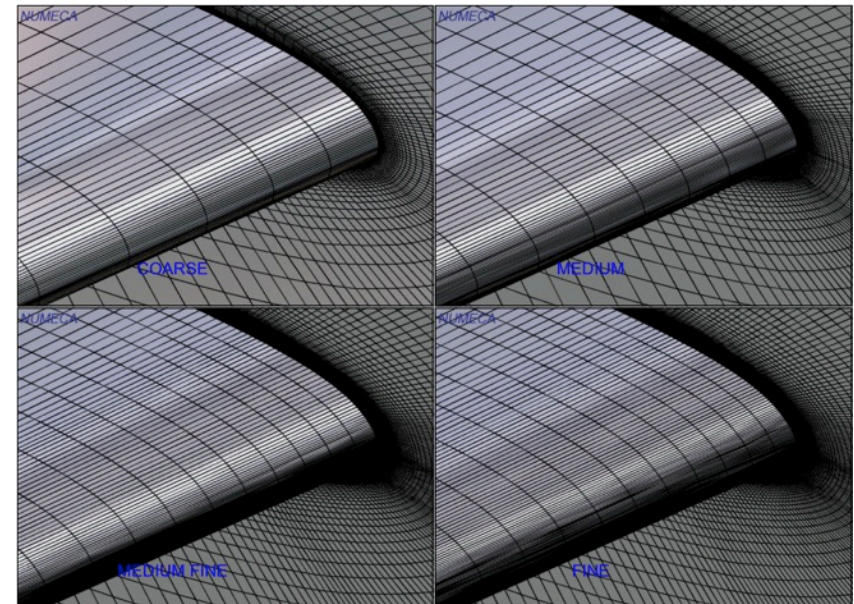
- robustness for all flow conditions, including unsteady and separated flows, for any fluid
- Reliability and validation are a main challenge, as most of the industrial configurations are outside the available validation range

## Our experience

- Density based approach, with efficient preconditioning for low speed and incompressible fluids
- Central schemes with scalar artificial dissipation (Jameson type AD) appear as the most robust, over the whole range of Mach numbers, including efficient agglomeration multigrid
- Are independent of the fluid state equation (unlike many upwind schemes)
- However, scalar dissipation leads to a higher mesh dependence,
- This is significantly reduced with matrix dissipation, which is however less robust



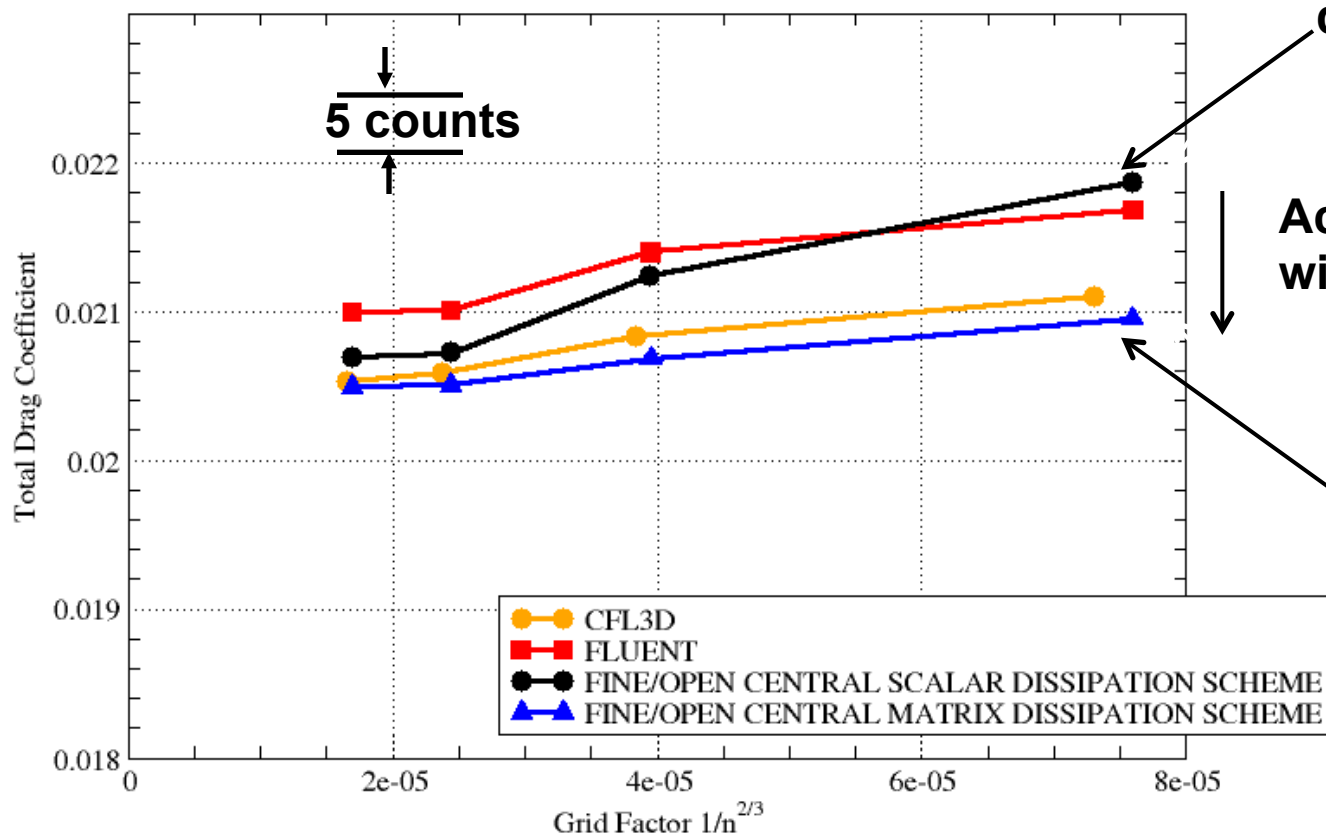
- **Mesh convergence study performed on Structured Multiblock grids from DPW3 website (Boeing grids constructed with ICEM)**
  - Coarse: 1,7M cells
  - Medium: 4M cells
  - Medium Fine: 8,3M cells
  - Fine: 14,3M cells
- **Meshes used by FLUENT, CFL3D and FINE/OPEN.**



# DPW3 results

## DPW3 - Wing W1

Grid convergence on structured for the total Drag coefficient



Scalar  
dissipation

Accuracy improvement  
with matrix dissipation

Matrix  
dissipation

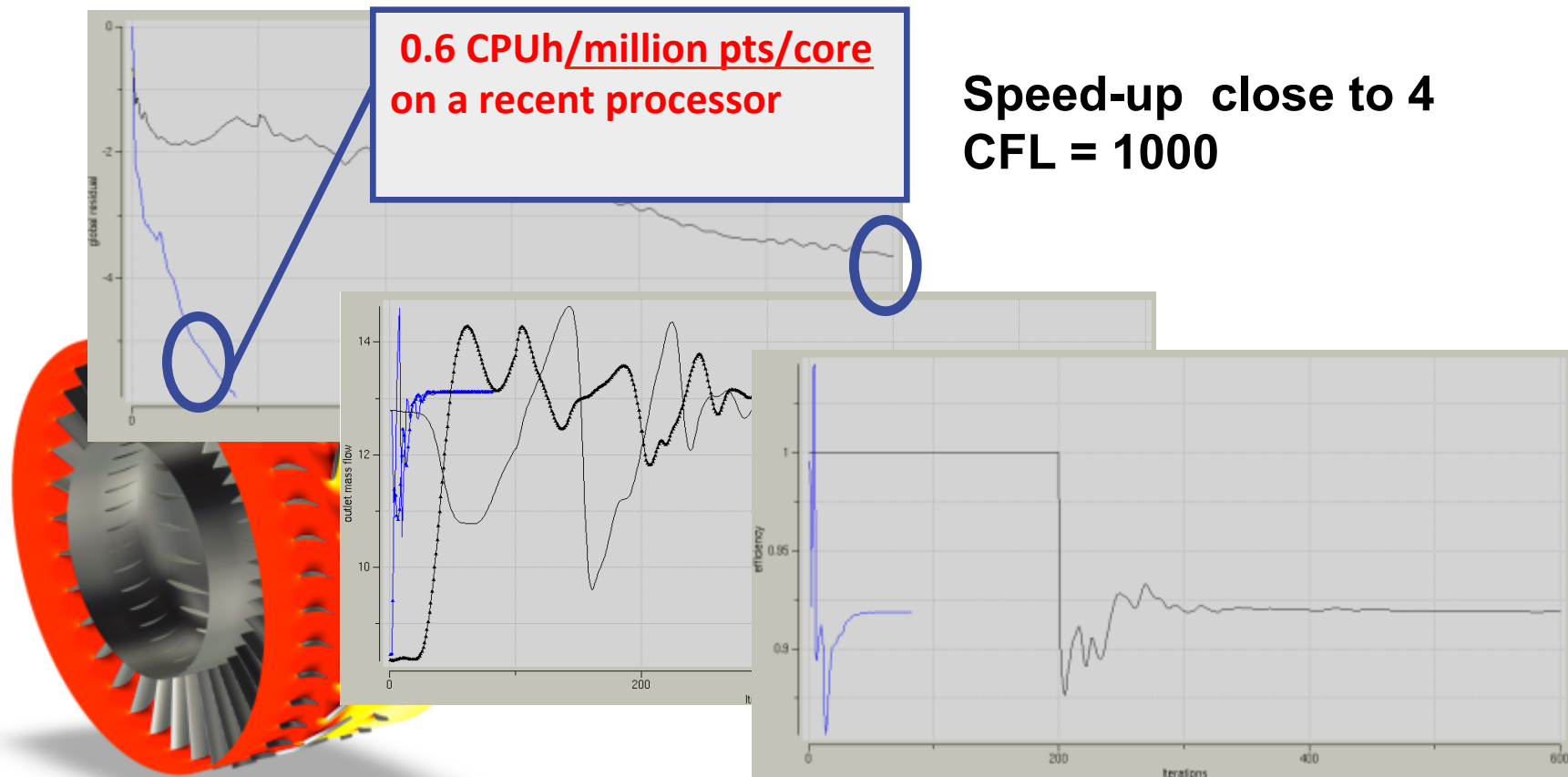


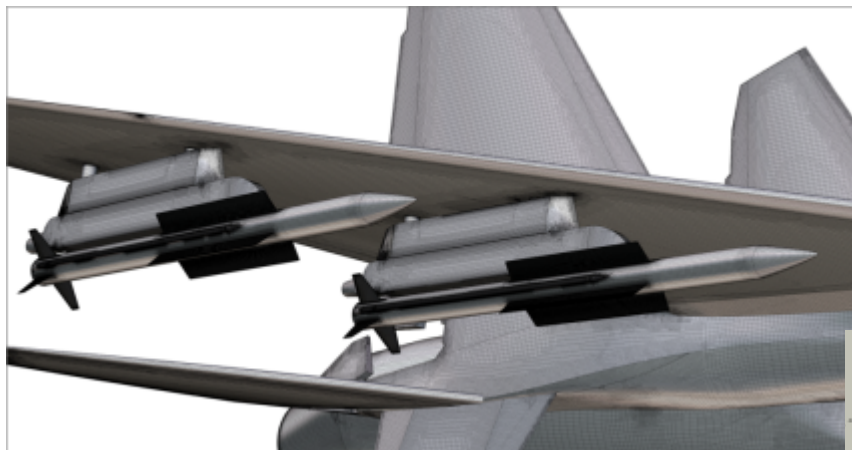
## Challenges on CFD tools

- ♦ RANS methods are to remain the reference in industry, for many years to come, particularly for design optimization
  - ♦ **Need to develop new and improved turbulence models to increase the accuracy and range of applicability of the models, particularly under separation**
  - ♦ Investigate the potential of algebraic and differential Reynolds stress models
  - ♦ Need to develop new and improved transition models, including surface roughness, for application to laminar flow control
- ♦ Methods have to be developed to reduce the CPU time for both steady and unsteady RANS simulations
  - ♦ For steady RANS, multigrid and combination of explicit and implicit time integration are to be investigated
  - ♦ For unsteady RANS, Reduced Order Modeling (ROM) techniques should be developed
  - ♦ For instance, a Non-Linear Harmonic method for unsteady rotor-stator interactions

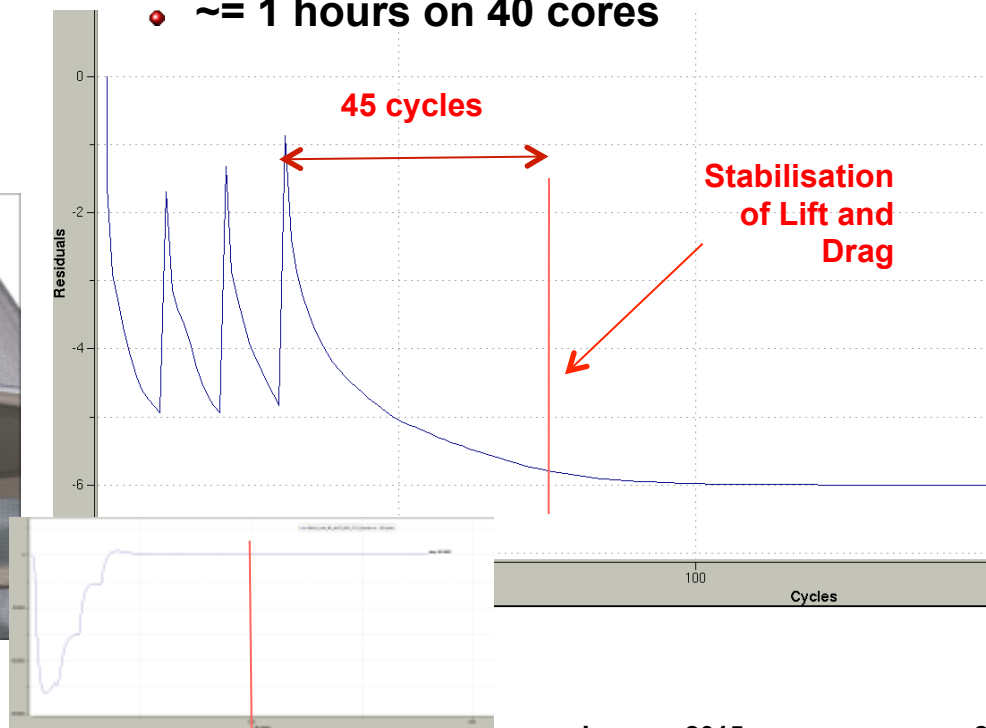


Application to a 1 ½ axial compressor stage with Fine™/Turbo  
 Meshed with Autogrid5™: 6.4 Mpoints





- **HEXPRESS 26 Million Mesh Points**  
(half configuration – Mach 0.8 - Angle of Attack: 10 deg – With stores – Turbulent NS)
- **Full convergence achieved in  $\sim 45$  cycles**
- **$\sim 1$  hours on 40 cores**





## ***NLH for unsteady flows***



# Non-Linear Harmonic (NLH) Method

## Casting in the frequency domain:

The unsteady conservative flow variable can be written as a time-mean plus an unsteady perturbation:

$$u(x, t) = \bar{u}(x) + \sum u'(x, t)$$

Each fluctuation  $u'$  is decomposed into N harmonics, the first being the fundamental (ex: associated with the Blade Passing Frequency (BPF) of an adjacent blade row).

To one periodic perturbation of frequency  $\omega$ , a complex amplitude is associated:

$$u'(x, t) = (\tilde{u}_R + i\tilde{u}_I) e^{i\omega t} + (\tilde{u}_R - i\tilde{u}_I) e^{-i\omega t}$$

$$\tilde{u} = \tilde{u}_R + i\tilde{u}_I$$

# Non-Linear Harmonic Method

Replacing the variables by their decomposition in the unsteady Navier-Stokes equations:

$$\frac{\partial U}{\partial t} + \sum F_c \vec{S} + \sum F_v \vec{S} + Q = 0$$

**Time -  
Averaging**

$$\sum \overline{F_c} \vec{S} + \sum \overline{F_v} \vec{S} + \overline{Q} = 0$$

**Linearization**

$$i\omega \tilde{U} + \sum \tilde{F}_c \vec{S} + \sum \tilde{F}_v \vec{S} + \tilde{Q} = 0$$

For both sets of equations, a pseudo-time derivative term is added in order to march in time to the solution.

# Non-Linear Harmonic Method

## Non-linear harmonic method

- The process introduces stress terms, analog to Reynolds stresses, called *Deterministic Stresses*, representing the full effect of the unsteadiness on the time-averaged flow
- NLH can be seen as a deterministic stress-based unsteady approach, where these stresses are obtained by the solution of the harmonic equations
- Provides a more accurate time-averaged solution, compared to mixing plane approaches
- Only one inter-blade channel is meshed per row.
- For each harmonic, the cost is equivalent to two steady state solutions, based on the existing steady flow solver.
- All unsteady flow properties can be reconstructed in time.
- Reduces the loss of continuity at the rotor/stator interface
- Provides a near-field tonal noise prediction



# NLH for Unsteady R-S interactions

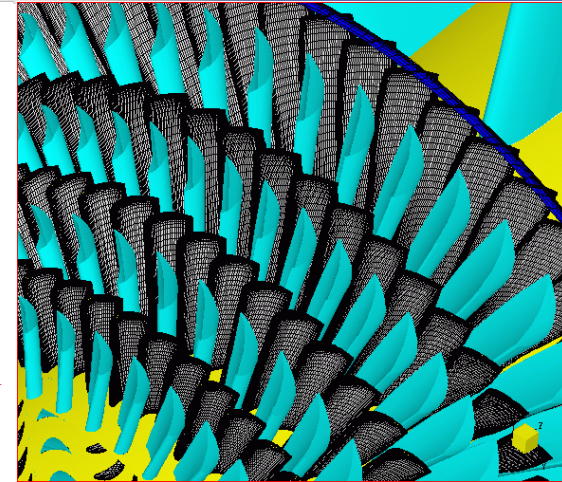
## Unsteady simulations URANS

- Reduced order models, such as Fourier decompositions

## Non Linear Harmonic Method

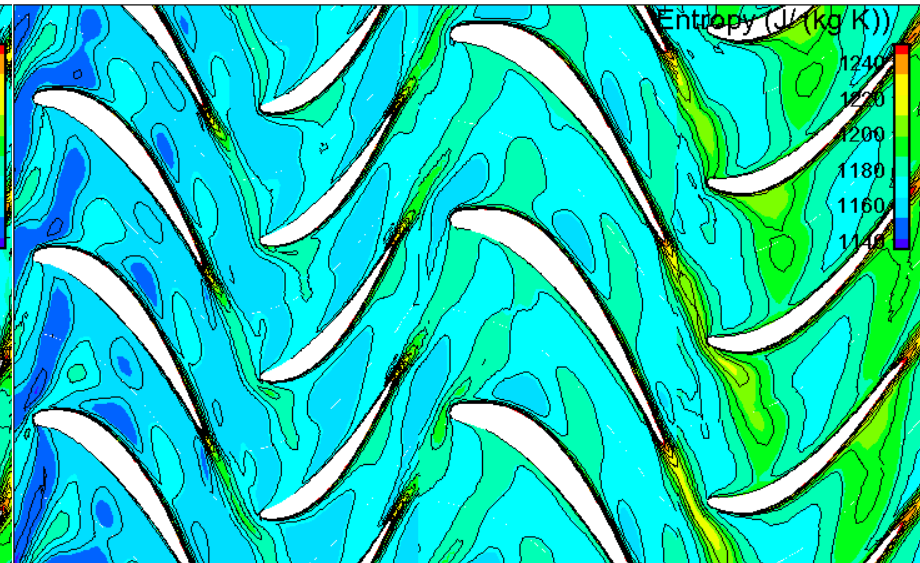
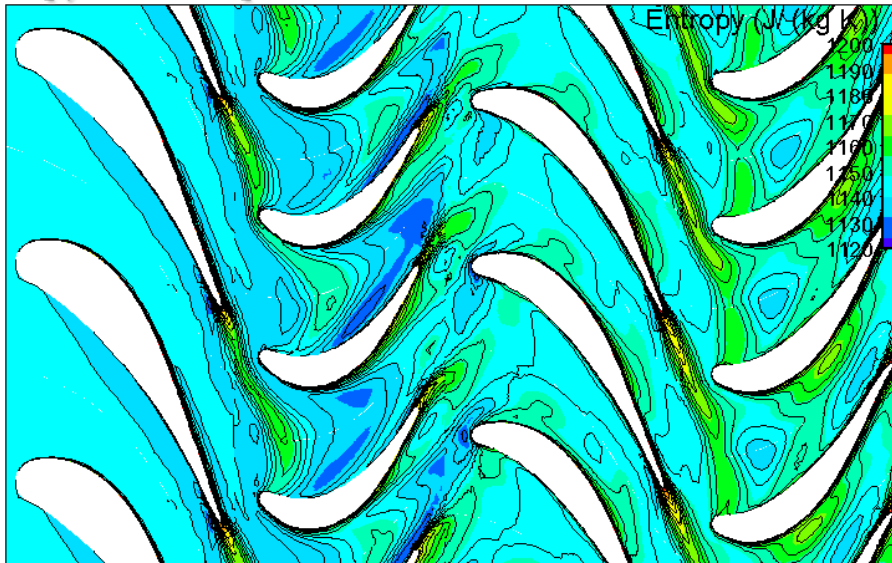
### Full 3d Unsteady Multistage Simulation: 4 stage turbine

7 M points - 6 hours CPU time on 16 processors, instead of 20 days for a full unsteady solution



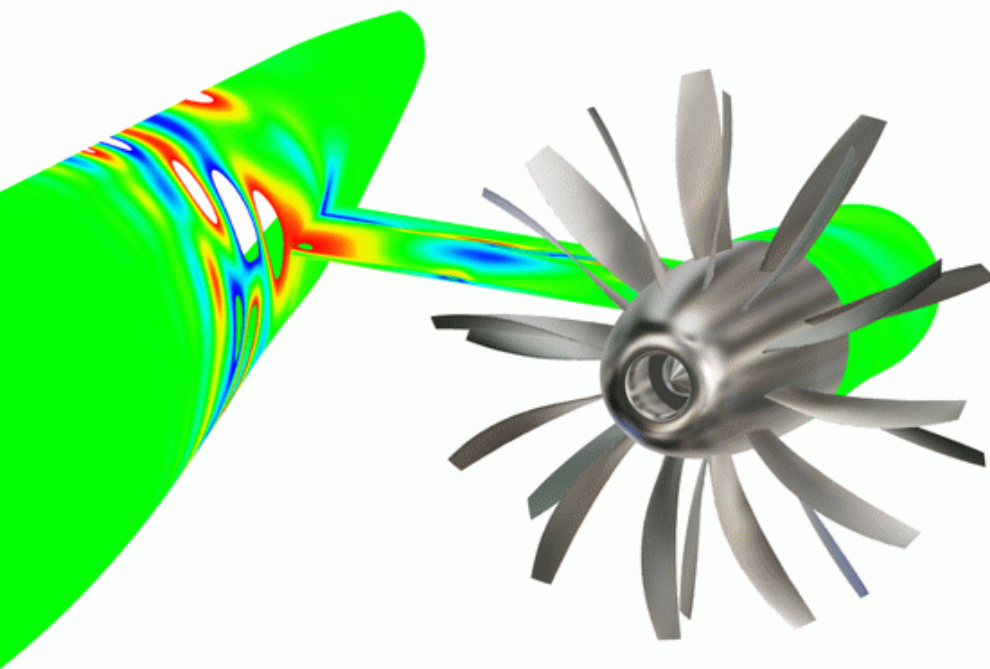
Entropy at 50 % span Stages 1 & 2

Stages 3 & 4



Compared to full sliding grid simulations, the gain in CPU time is close to 2000

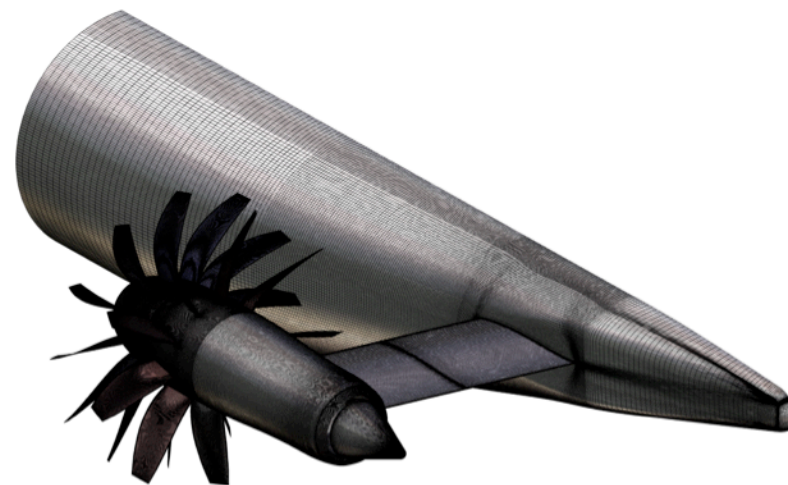
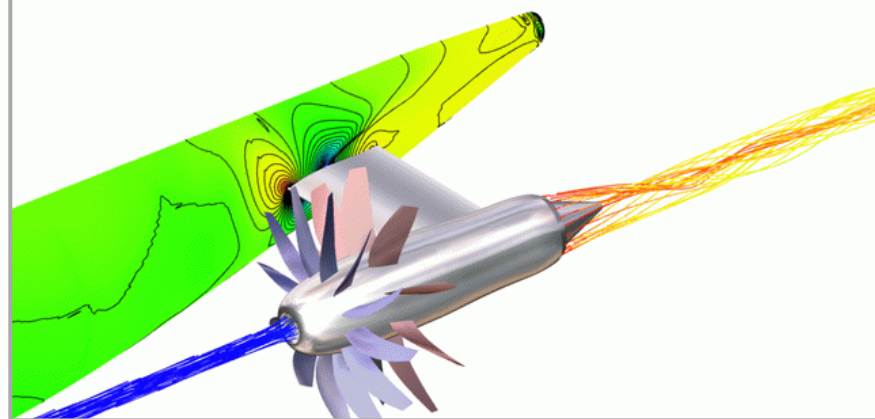
0001/360



Animation of near field acoustic signature on fuselage, pylon and nacelle

From NUMECA Int. and Rolls Royce D

0001/360





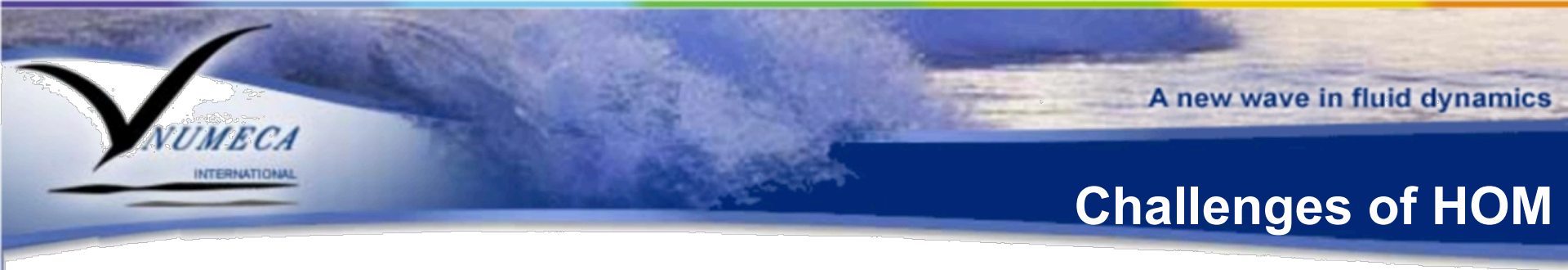
# ***LES and High Order Methods (HOM)***





## High order methods (HOM)

- **Current CFD codes are nominally of second order**
  - Valid strictly on Cartesian grids
  - On unstructured grids there is a general loss of accuracy due to irregular cell shape and sizes
  
- **High order methods (HOM) - DG**
  - Arbitrary order of accuracy
  - Provides highly accurate solutions on coarse grids
  - Applicable to unstructured grids, and every cell type.
  - Local high-order reconstruction – Highly parallelizable
  - A DG code is present in FINE/Open, developed in partnership with TsAgi (Dr A. Wolkov group) but not yet available at industrial level
  - A new “Flux Reconstruction” (FR) method is being developed



- **Necessity for high quality curved meshes**
- **Future potential**
  - From the IDIHOM project (see Presentation at this conference by N. Kroll et al.), it appears that HOM might not be competitive compared to current efficient finite volume CFD codes, for steady RANS
  - But room is still available for performance improvements
- **However, HOM are highly competitive for unsteady flows, in particular for CAA and LES/DNS**

# Spalart 2012 vision for LES/DNS on industrial relevant configurations

Spalart, June-August 2012



- We believe this estimation to be too pessimistic
- It is based on standard second order schemes
- HOM offer new opportunities, which should be exploited

## Spectrum of Approaches to Turbulence

Name	DNS	LES	DES	RANS
Empiricism	No	Low	Medium	High
Unsteady	Yes	Yes	Yes	No (can be)
# of points (Boeing wing)	$10^{20}$	$10^{11}$	$10^7$ to $10^8$	$10^7$
In Service (Boeing)	2080*	2045*	2010 (sub-regions)	1995
Vibration, Noise	Yes	Yes	Yes	No (buffet maybe)

\*Assuming Moore's Law holds!





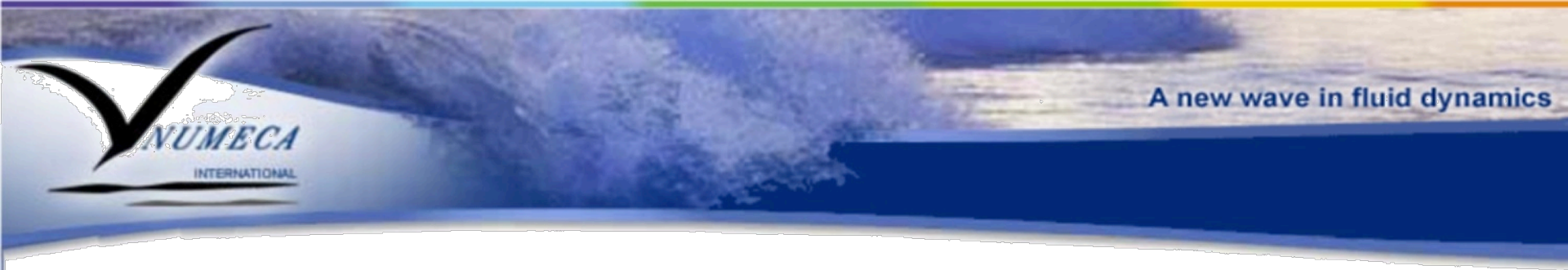
# The TILDA project

## Objective:

- High level LES (DNS) at industrial level in the near-term (3-5 years)

## Apply:

- High order methods on unstructured grids for industrial relevant configurations
  - Including advanced grid generation methods for curved grids on complex configurations
- Level of several billion points or degrees of freedom
- Run massive parallel computations on 50,000+ cores,
  - This level of parallel cores should be available to industrial designers for daily use in the near-term
- Turnover time: one or two days!!!
- **Submitted Horizon 2020 Project led by NUMECA**
  - Waiting (hopefully) for EC confirmation



# *Uncertainty Quantification (UQ)*

# Challenges in Uncertainty Quantification (UQ)

- UQ is nowadays recognized as an important component of CFD and multidisciplinary simulations
- This is the objective of the current UMRIDA (Uncertainty Management for Robust Industrial Design in Aeronautics) EU project, led by NUMECA

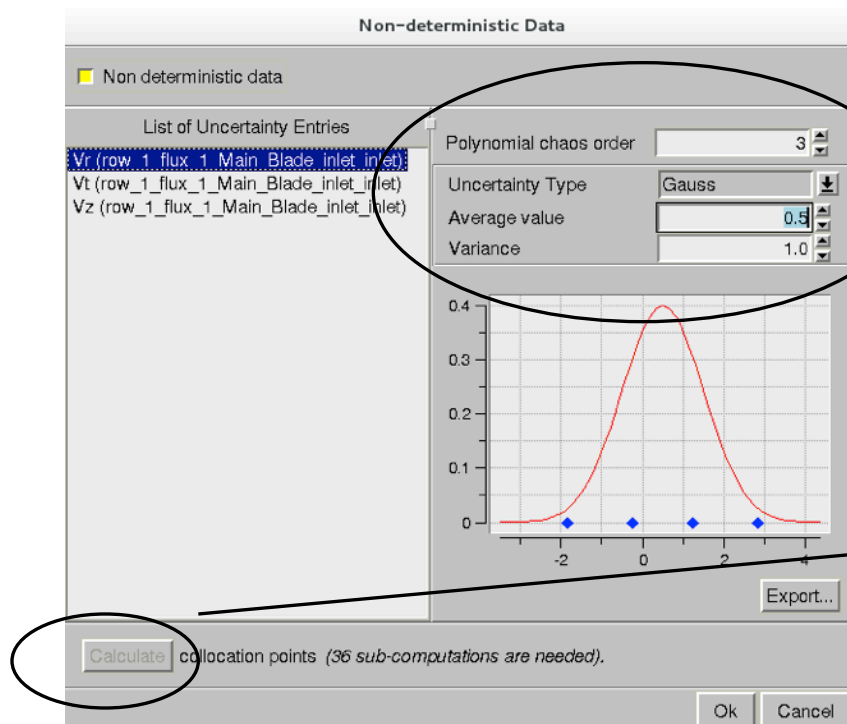
## NUMECA UQ Module

- Based on Polynomial Chaos collocation method
  - Operational uncertainties: stochastic definition of full profiles
  - Geometrical uncertainties
  - Prediction of output pdf's
  - Requires high level of automation (geometry) and user-friendliness
- 
- UQ module is introduced on this basis, with automatic parameterization of geometry, meshing and simulation set-up



## Handling operational uncertainties – Automatic generation of computation

- ❑ Tensor-product or Sparse Grids are used to calculate entries for sub-computations



Define type and values of PDF to be imposed through a dedicated GUI window

Calculation of collocation points and automatic generation of individual non-deterministic simulations:

- Tensor-product
- Sparse Grids

# Application to Rotor 37

## Test case description: Rotor 37

❑ Detailed description of geometry, exp. set-up and a series of simulations cross-plotting the predictions can be found in [Dunham (1998)]

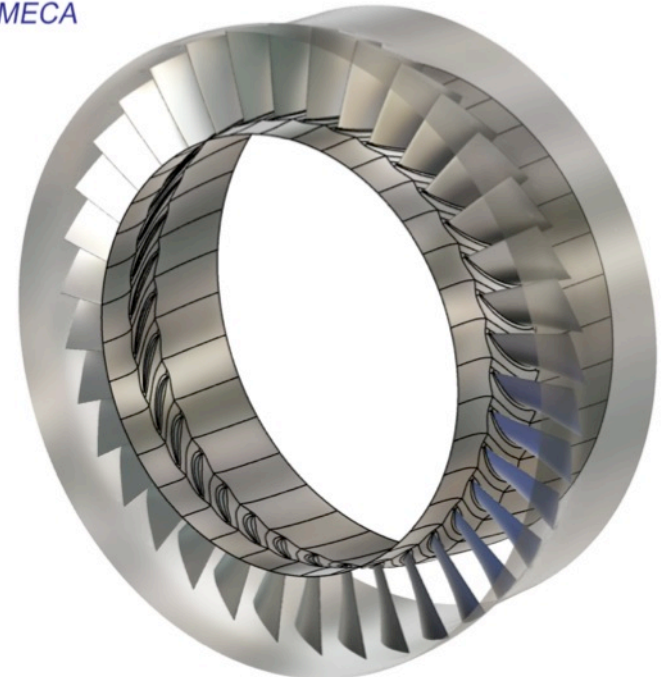
❑ Test case and UQ model

- ❑ Mesh size: 2 639 973 and 4 702 629 cells
- ❑ RANS + Spalart-Allmaras
- ❑ Rotating Hub: 17188 rpm

❑ Uncertainties: all PDFs are Gaussian

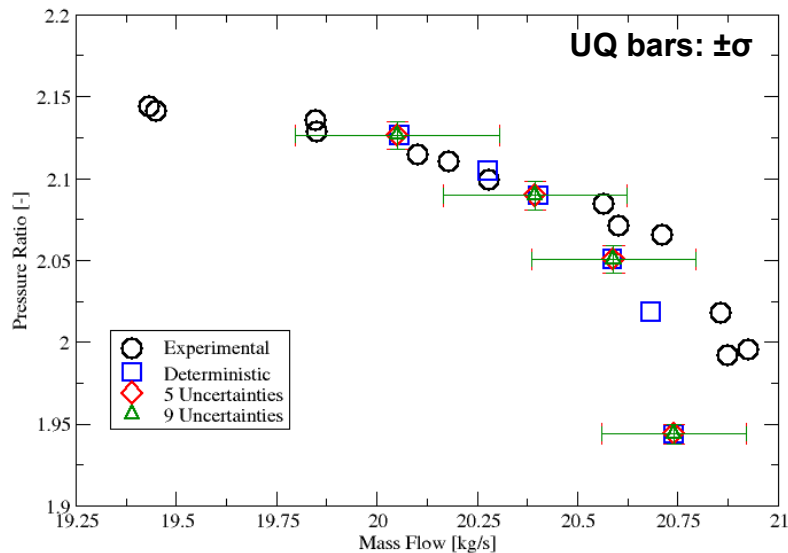
- i. 5 uncertainties: total inlet pressure, static outlet pressure, tip gap, LE angle, TE angle
- ii. 9 uncertainties: total inlet pressure, static outlet pressure, tip gap, LE angle (3 sections), TE angle (3 sections)

NUMECA

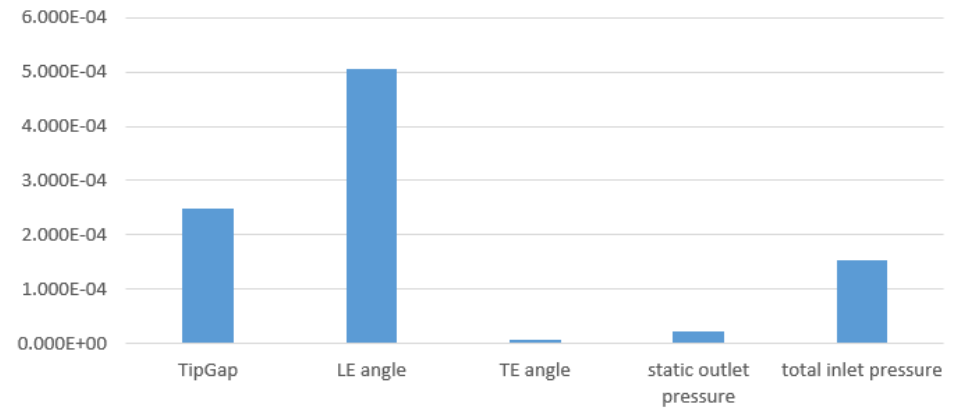


## Scaled sensitivity derivatives

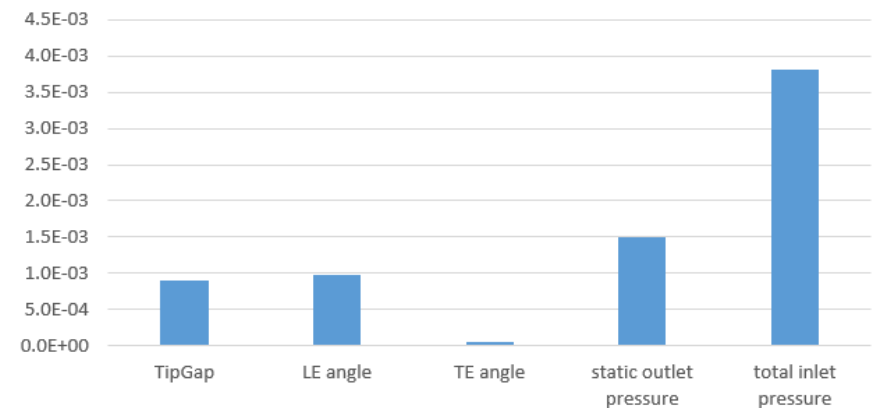
- Sensitivity derivatives allow to assess influence of a given uncertainty on the non-deterministic output
- Total inlet pressure most dominant for pressure ratio and mass flow rate
- Most influential parameter for efficiency is the LE angle



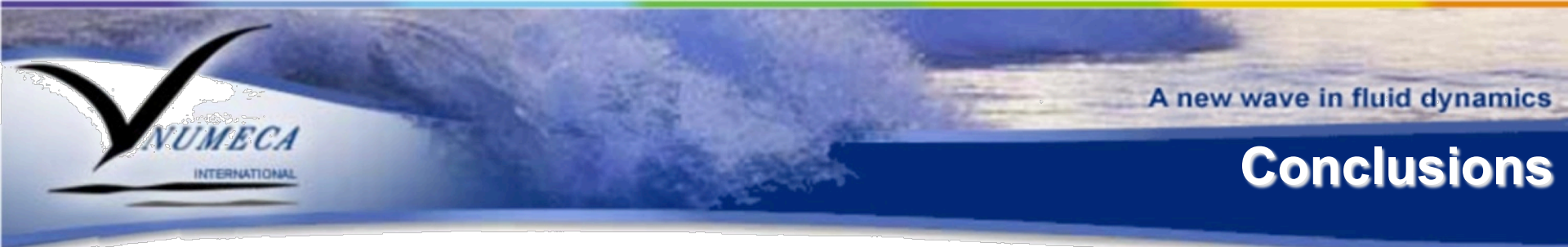
Efficiency scaled sensitivities - 5 unc



Pressure Ratio scaled sensitivities - 5 unc

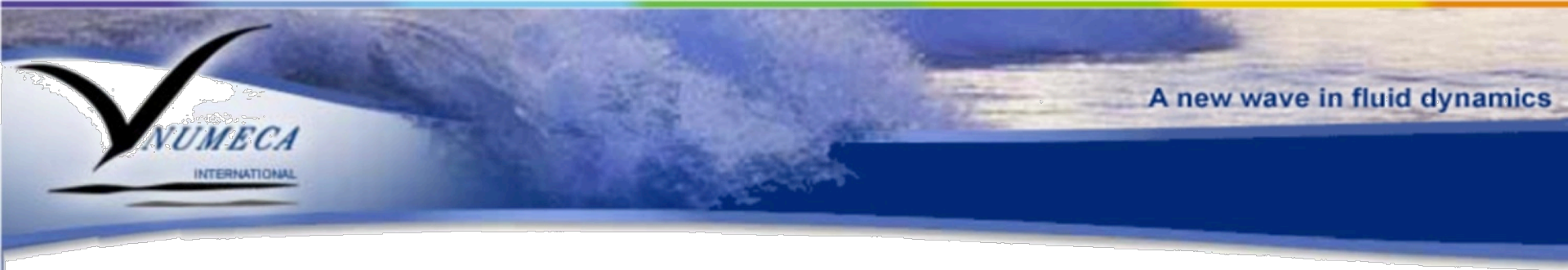






## Major challenges for Industrial CFD analysis and design software systems still lay ahead

- Efficient full automatic grid generation systems and flow solvers are to be developed further, particularly for very complex geometries
- The necessity for improvements in physical modeling, in particular turbulence and combustion models
- Fast, full parallel, CFD algorithms are required to reduce design cycle times
- The development of robust design methodologies taking into account the presence of uncertainties.
- **Next generation of industrial software systems requires high levels of integration of pre- and post-processors, with CFD/CHT/FSI solvers within a global optimization environment, with highly effective GUI's to minimize the engineering time associated to simulations and design**



**Thank you for your attention**

